

## Validation of bucket flow simulation using dynamic pressure measurements

Author	Firm / Institution	City, Country	Lecturer (x)
<i>Reiner Mack</i>	<i>Voith Siemens Hydro Power Generation</i>	<i>Heidenheim, Germany</i>	x
<i>Dr. Thomas Aschenbrenner</i>	<i>Voith Siemens Hydro Power Generation</i>	<i>Heidenheim, Germany</i>	
<i>Dr. Wolfgang Rohne</i>	<i>Voith Siemens Hydro Power Generation</i>	<i>Heidenheim, Germany</i>	
<i>Dr. Mohamed Farhat</i>	<i>Swiss Federal Institute of Technology</i>	<i>Lausanne, Switzerland</i>	

### Abstract

The paper below is a follow up of the authors contribution to the IAHR 2002, in which the simulations of the flow through Pelton buckets were presented. In 2002, a good consistency between the predicted and measured efficiency was presented.

This paper will first present a discussion of several possible strategies for bucket flow simulation. It will be described, why the authors favour the 3 bucket approach instead of a periodic simulation.

The second part describes the detailed validation of the simulation with results of a dynamic pressure measurements campaign carried out in the rotating buckets. It will be shown, that the predicted time dependent pressure trends are in good agreement with the measurements. Both, the shape and the height of the development are well predicted. Even the situations where values below ambient pressure occur are in remarkable agreement.

### Résumé

Le présent papier est une suite de la contribution des auteurs au symposium AIRH 2002 dans laquelle les simulations de l'écoulement dans les augets Pelton ont été présentées. En 2002, une bonne concordance entre la prédiction et la mesure du rendement a été réalisée.

Dans la première partie de la présente étude, nous présentons une discussion des différentes stratégies possibles de simulation. Nous justifierons notre choix de l'approche de simulation 3-augets au détriment de l'approche de simulation périodique.

La seconde partie est dédiée à la description détaillée de la validation de la simulation numérique à l'aide de mesures embarquées de champs de pression dynamique dans les augets. Nous démontrerons que l'allure et l'amplitude, prédites pour la pression sont en bon accord

avec celles mesurées. Un accord remarquable est ainsi obtenu même dans le cas où la pression est inférieure à la pression ambiante.

## Introduction

Because of their complex flow process, Pelton turbine designs are mainly based on the longstanding experience and experimental developments carried out in the hydraulic laboratories. Design support with Computational Fluid Dynamics (CFD), similar to reaction turbines (Aschenbrenner et al. Ref. 1), was in the past only possible for the distributor section of the turbine, where the flow is single phased and stationary.

The free surface parts of the turbine, like the jet development and the flow through the rotating buckets had to be analyzed by traditional design methods as described by Brivio (Ref. 2.). A first step into the computerized approach for solving the flow through the buckets was the introduction of the animated cartoon method by Kubota et al. (Ref. 3).

The numerical simulation of the flow through Pelton buckets by taking into account the viscous terms of the Navier Stokes equations, however, is one of the high end applications within hydraulic turbo machinery. Its unsteady nature, together with the multiphase system and moving geometry makes it a challenge for every CFD code. In 2002 the first successful bucket flow application utilizing the Volume of Fluid method and moving grid features of a commercial code were presented by Mack et al. (Ref 4). At that time, the simulated angular momentum and its derived efficiency showed a good agreement with model test results. Moreover, the development of the flow, visualized by the phase interface between water and air compared very good to the typical observations made on the model test rig.

However, in order to ensure that the pressure development predicted by CFD is reasonable, detailed pressure measurement at several locations on the buckets was set up and compared to the simulation data. The result of this validation together with a discussion of the possible simulation approaches is shown in the following sections.

## Discussion of simulation approaches

Unlike to the common approach for reaction turbines, the simulation of a Pelton runner cannot be reduced to a steady state problem without neglecting essential mechanisms of the energy transfer. The flow through the buckets has to be simulated for the complete working cycle of at least one bucket of the runner. Subsequent analysis can then be done either by investigating snapshots at different times during the working cycle (water sheet size and motion, pressure distribution, angular momentum generated) or by looking onto the whole cycle with an animated analysis of all time steps simulated. For a quantitative analysis also integral values can be build.

This simulation requirement, however, allows different solution approaches that can be followed, ranging from the complete runner as one extreme to just one bucket as the other. In the following, approaches that are reasonable will be discussed in terms of their computational requirements and analysis capabilities. Figure 1 shows the setups that are discussed.

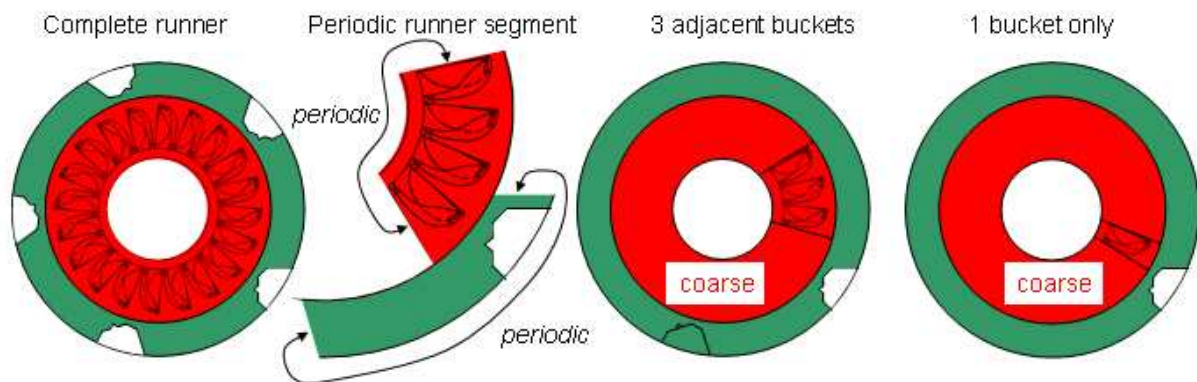


Figure 1: Comparison of possible simulation approaches

### Complete runner

The simulation of the complete runner allows the most flexible analysis. The number of buckets and the number of jets driving the runner is completely independent. From bucket design to jet interference analysis all simulations are possible. By enlarging the stationary part to the housing dimensions even simulation of the flow in the housing is possible, provided that the gravity force is active and the grid is refined to the required minimal size. In this case, a horizontal axis machine would require one half of the runner to be modelled, whereas for a vertical turbine the complete turbine housing and runner would have to be included into the computational model.

The downside to this approach, however, is obviously the huge number of grid points required and with it the tremendous computational effort. Therefore, for the industrial use, where the design of new hydraulic bucket profiles is the first reasonable application, this approach is not practical. It has its justification if the interaction of the runner and housing should be investigated, however, a true design optimization of the housing is still not possible with today's computational capacity.

### Periodic runner segment

A reasonable reduction of the computational effort is the assumption of a periodic flow regime, which is typical for vertical axis machines. It allows nearly all the analysis possibilities as the complete runner approach, with the exception of the housing simulation for horizontal axis turbines.

A very important prerequisite, however, is, that the code used for the analysis allows different periodic segmentation angles for the rotating and the stationary part of the grid. Otherwise the analysis is only limited to certain combinations of bucket numbers and numbers of jets. Runners holding 19 or 23 buckets are not possible to be analysed if this requirement is missing. For the hydraulic design, the number of buckets is an essential parameter. The difference in the momentum development generated by one bucket for an analysis of 18 and 19 buckets is shown in Figure 2. Also the risk to jet interference depends strongly on the number of buckets. Therefore, this approach is only reasonable, if the code allows different segmentation angles.

### 3 adjacent buckets

A further significant reduction of the required computational effort is the simulation of 3 adjacent buckets. This approach generates for the middle bucket the conditions that are common to all buckets of a runner at any given time. Therefore all mechanisms and interactions that take place during the working cycle is considered, which make this approach a good support to the hydraulic bucket design. Even the jet interference simulation is possible, if a second jet is defined that will impinge on the 3 buckets at a later point in time.

A possible disadvantage to this approach is that if the utilized code requires a closed sliding interface, the rotating part that is not of interest for the analysis needs to be filled with a coarse grid. Furthermore, the 3 bucket approach is not able to analyse the flow in the housing directly. However, the angles of the water sheets at the outlet of the bucket can be used for further analysis that support the design of the main housing parameters or modifications to the housing inner parts.

### 1 bucket only

The least computational effort is required if only one bucket is simulated. This approach is capable of supporting the hydraulic design of the inner bucket profile with a very fast turn around time. However, assumptions during the simulation have to be made for the cutting of the jet by an imaginary following bucket as done by Janetzky (Ref. 5). Also the same downside as for the previously described approach may occur, if the utilized code requires a closed sliding interface.

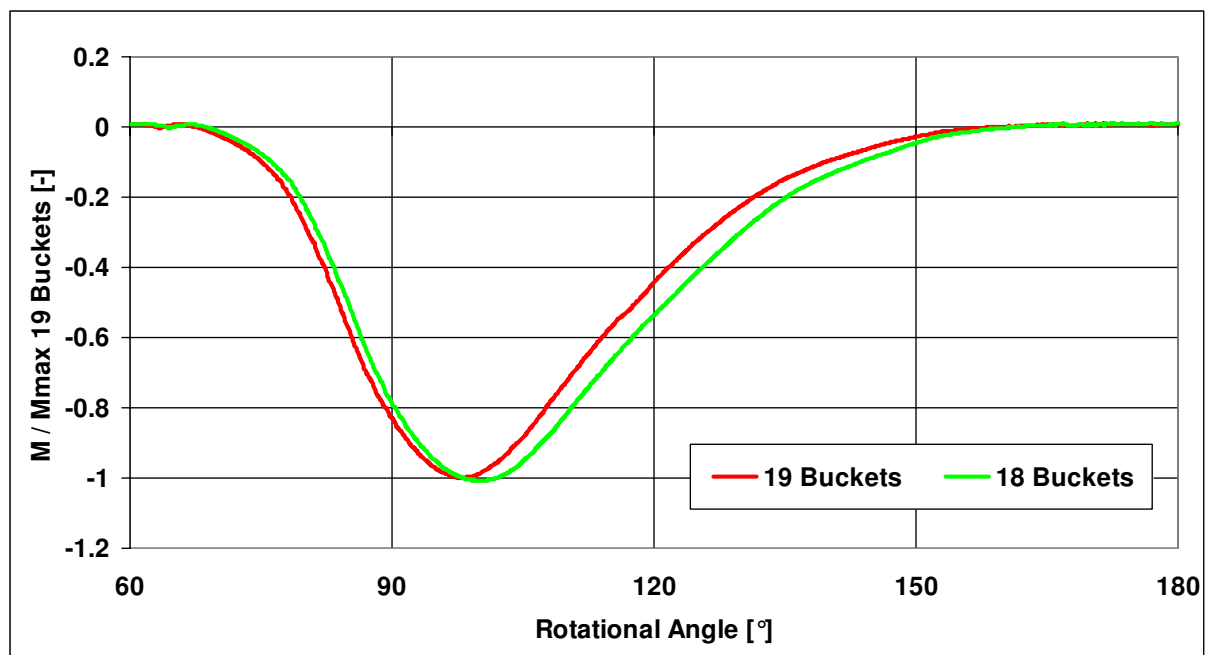


Figure 2: Angular momentum for 18 and 19 buckets, predicted by 3 bucket approach

## Validation with transient pressure measurement

Figure 3 shows the development of the angular momentum for the front and backside of one complete working cycle of a model bucket predicted by CFD. As expected, one can see, that the front and back side of the bucket has its part onto the energy conversion, which means a complete analysis needs to take into account both, the hydraulic active parts on the front and backside of the buckets. Because of this, the validation described in the following is done with measurement locations taken on the front and backside of the bucket.

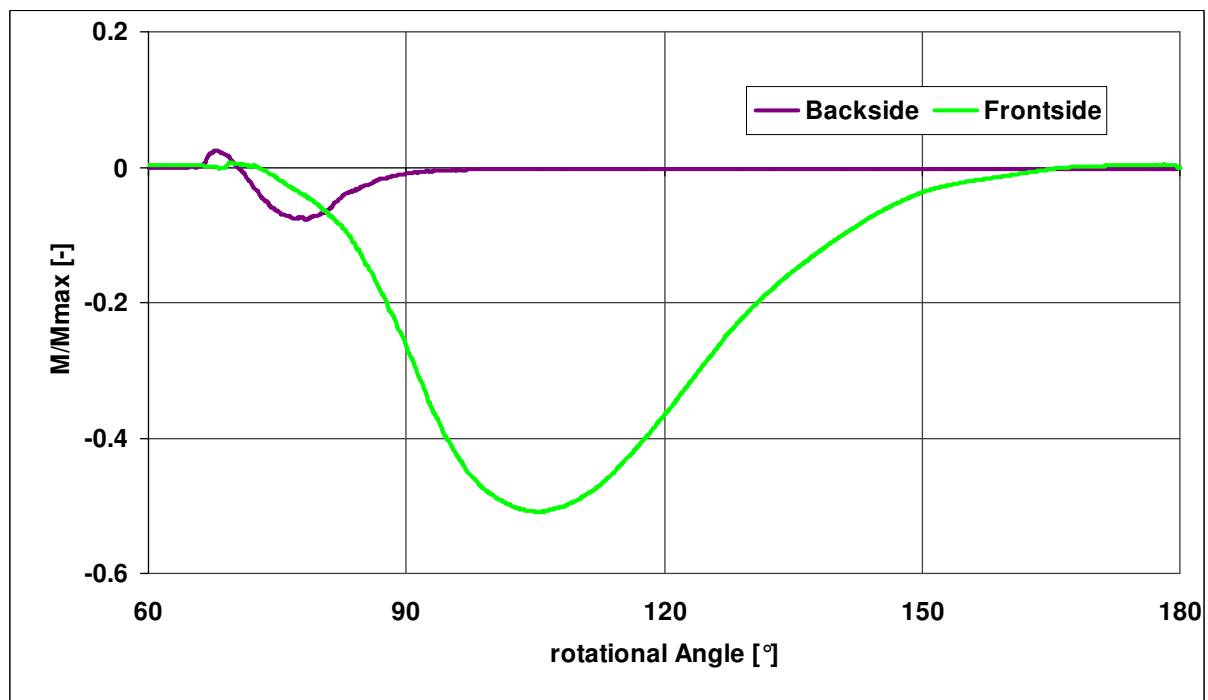


Figure 3: Angular momentum generated by front and backside

## Experimental configuration and measurement

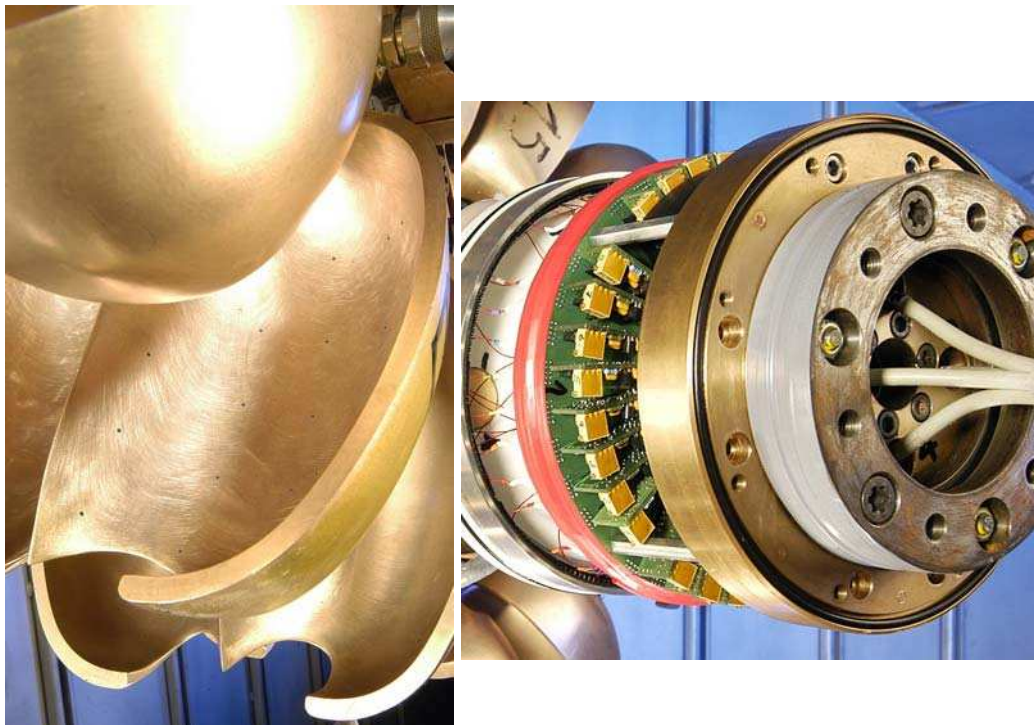
The experimental setup used for this measurement has been developed by the IMHEF in close cooperation with Voith Siemens Hydro Power Generation. It is similar to the one used successfully for Francis and Pump Turbines presented by Farhat et al. (Ref. 6).

32 miniature pressure sensors of 1mm diameter have been mounted into the front and back side of two model buckets. Figure 4 shows the sensors distributed over the inner side of the bucket. A new technology, making only 2 mm material thickness necessary was applied, allowing the sensors to be placed very close to the cut out edge.

The wires of the sensors are led through cable paths into the basement of the bucket and connected there to the signal conditioning unit through a flexible pipe. Figure 5 shows a picture of the unit mounted onto the runner shaft. It holds for each of the 32 channels a

separate pre amplifier and an anti aliasing filter allowing amplification factors ranging from 1 to 1000.

Connected to the conditioning electronic are the acquisition boards located also on the turbine shaft. Each of these boards hold four recording channels together with four 12 bit A/D converter. The maximum sampling frequency is 20 kHz and the installed memory capacity allows the storage of 64k samples per channel. A host computer for data transfer and analysis is connected to the shaft over a slip ring connection.



*Figure 4 (to the left) Bucket instrumented with miniature pressure sensors.*

*Figure 5 (to the right) Signal conditioning unit mounted onto the runner shaft.*

The pressure sensors were calibrated statically using a special pressure vessel. The absolute error for the calibration is within 0.25 % of the maximum measured pressure. The angular position and the rotational speed of the runner is detected by a magnet inductive trigger signal, which also controls the phase averaging of the measured pressure signals.

About 40 to 60 cycles, depending on the  $n_1$  got recorded in one measurement shot. The raw data were then phase averaged resulting in a time dependent average pressure development for each sensor. This pressure development is then used for the comparison with the simulation data.

### Validation results

In Figure 6 a comparison of the time dependent pressure signals for five sensor locations of the bucket inner side is presented. Both, the simulated and measured pressure data have been normalized with the maximum pressure value measured for the specific sensor. The

comparison was performed for an operation point close to best efficiency. For the simulation the above described 3 adjacent bucket approach was used.

When comparing the measurement results with the simulated pressure development it can be seen, that the general trend is well predicted. The shapes of the time dependent curves for the sensors in the front side are in very good agreement. A slight under prediction occurs at all locations in the front side, which is basically the cause for predicting the efficiency about 2 to 3 % lower than the measurement.

The situation on the backside of the bucket is very well predicted. A comparison for two sensor locations on the backside of the bucket profile is shown in Figure 7. Here the same normalization as for the front side sensor data has been applied. When comparing the data, one can clearly see, that the shape and height of the sensor locations are in remarkable agreement.

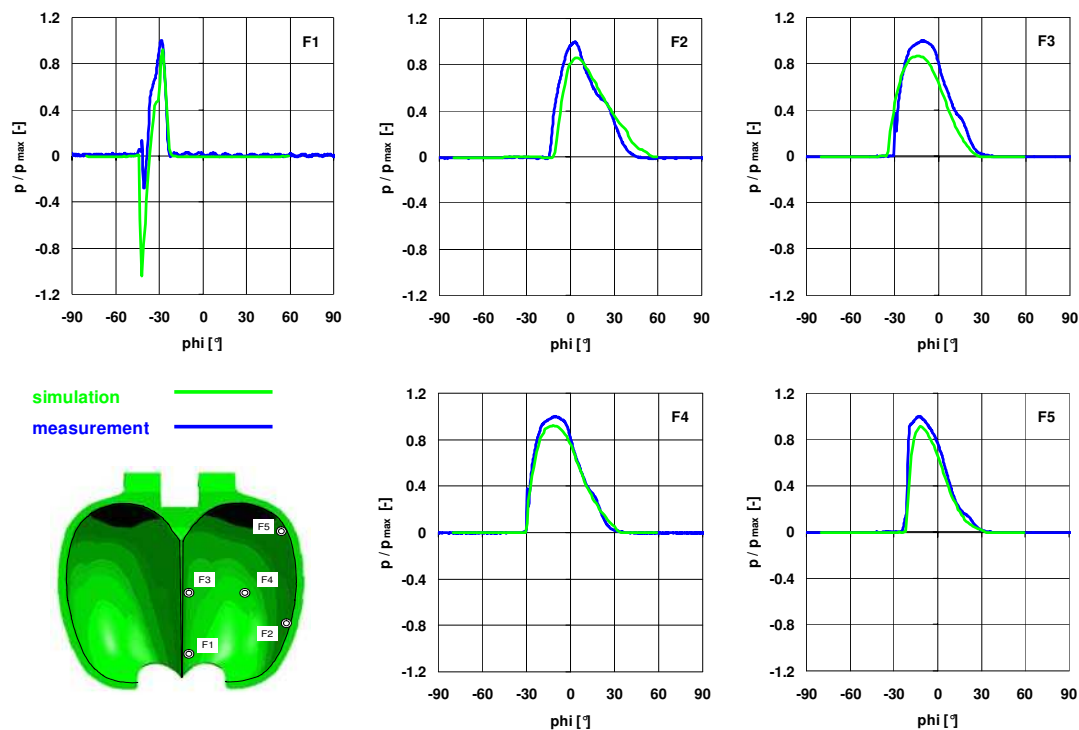


Figure 6 Pressure development for sensors at bucket inner side



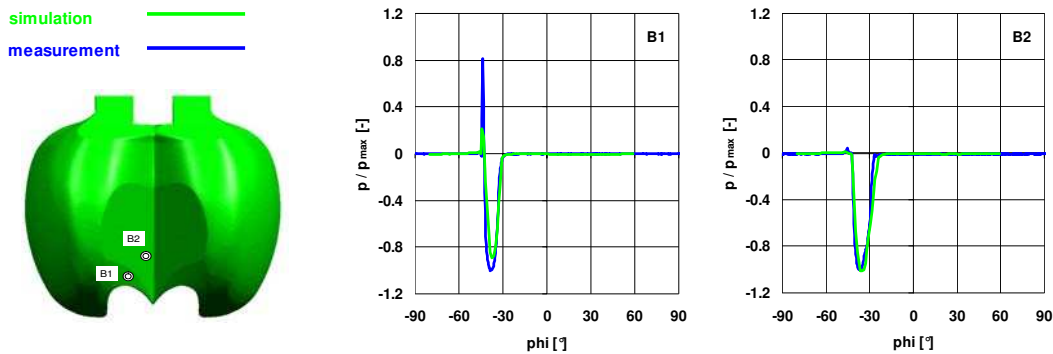


Figure 7 Pressure development for sensors at bucket back side.

## Conclusion

The presented validation shows a good agreement between the simulated and measured pressure development. Especially the situation on the backside of the buckets, where values below ambient pressure are dominant is in very good agreement. This proves, that the 3 adjacent bucket method achieves the necessary accuracy for supporting the bucket design. Together with the flexibility in the number of buckets and the reasonable computational effort, the realistic modeling of the essential mechanisms during the cycle makes this set up the most promising approach when it gets to the design of improved hydraulic profiles.

## References

- Ref. 1 T. Aschenbrenner, M. Goehringer, W. Moser, 2000, "Numerical and experimental flow analysis in a Kaplan Turbine", *Proceedings of the 20<sup>th</sup> IAHR Conference on Hydraulic Machinery and Cavitation, 2000*, Charlotte, USA, CFD-F13.
- Ref. 2 R. Brivio, 1995, "Uprating and refurbishing of Pelton Hydro Powerplants in Italy", *Proceedings of Uprating & Refurbishing Hydro Powerplants V*, 9-11 October 1995, Nice, France, vol. 1, 11 pages.
- Ref. 3 T. Kubota, X. Jinjong, H. Takeuchi, J. Masuda, Y. Nakanishi, 1998, "Numerical analysis of free water sheet flow of Pelton buckets", *Proceedings of the 19<sup>th</sup> IAHR Symposium*, 1998, Singapur.
- Ref. 4 R. Mack, W. Moser, 2002, "Numerical Investigation of the flow in a Pelton Turbine", *Proceedings of the 21<sup>st</sup> IAHR Symposium on Hydraulic Machinery and Systems*, 9-12 September 2002, Lausanne, Switzerland.
- Ref. 5 B. Janetzky, 2001, "Ein Verfahren zur Berechnung instationärer Strömungen mit freier Oberfläche", *Dissertation Universität Stuttgart*, 2001.
- Ref. 6 M. Farhat, F. Avellan, U. Seidel, 2002, "Pressure fluctuation measurements in Hydro Turbine models", *Proceedings of the 9<sup>th</sup> International Symposium on transport phenomena in dynamic rotating machinery*, Februar 2002, Honolulu, Hawaii.